Die Cost

The following items are reminders of how die cost affects stock layout:

1. Die cost may be higher for higher productions.
2. Cutting more than one at a time results in higher die cost.
3. Double-pass dies are less expensive than cutting two at a time.
4. When extremely complicated blank shapes are to be cut, cutting more than one at a time would result in excessive die costs.
5. When extremely accurate blank sizes must be maintained, cutting more than one at a time would result in excessive die costs.
6. For simple round or square-edged blanks, multiple cutting at one time is often practical.

When making the stock layout, the designer must decide which is preferred—more operator time per blank or more machine time per blank.

The layouts shown in Figure 181 are for simpler two- or three-station progressive dies. Notice that all stock layouts are for single-pass operation. Progressive stock layouts are dimensioned in a manner similar to those presented in this section.

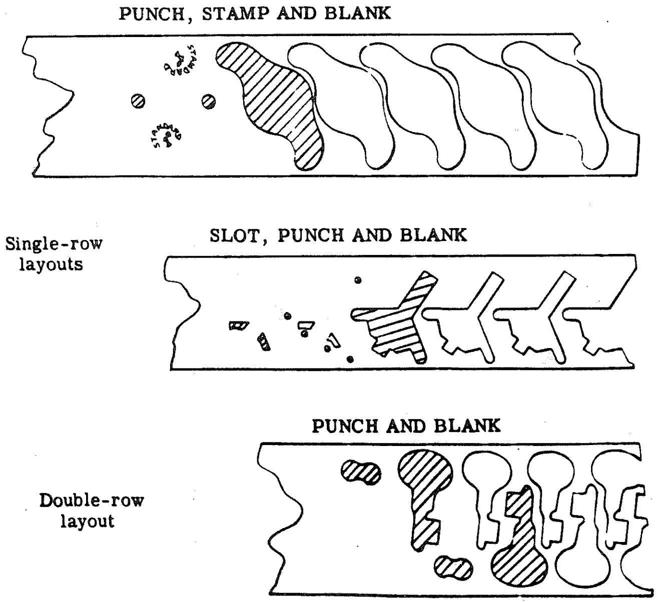


Figure 181. Progressive Stock Layouts

15

Processing of Sheet Metal Parts

The role of the process engineer in the over-all plant organization will be described briefly so that his function and responsibilities will then be more easily defined. The functions of a process engineer place him in direct contact with both the product engineering and production departments. Functions of product engineering include:

Product Engineering Functions

A. To design the product for function.
    1. Build models for testing.
    2. Provide part prints.
        a. Physical dimensions.
        b. Material.
        c. Special processes required, such as painting, plating, heat treatment, testing, etc.
    3. Provide tool design and construction aids.
        a. Master layouts.
        b. Templates.
        c. Master models.
    4. Provide specification manuals.
        a. Material specifications—chemical analysis and physical properties.
        b. Specifications for special processes—chemical and physical properties for plating, painting, heat treatment, etc.
        c. Procedures for testing and inspection.
        d. Procedures and specifications for joining processes such as welding, brazing, soldering, riveting, cementing, etc.
        e. Specifications for threads, gears, splines, keys, etc.

B. To design the product for customer satisfaction.

1. Sales appeal.
    a. Appearance—color and styling.
    b. Improvements—changes and additions over the old model.
    c. Designs to meet the needs of the customer.
2. Durability and life expectancy of product—parts are designed to satisfy customer by giving the durability expected in relation to cost.

C. To design the product for cost.
1. Cost of product must be low enough to compete with similar parts.
2. Cost must be high enough to provide a profit desired by owners of company.
3. Cost must be in correct relationship with durability and life expectancy.

D. To design the product for ease of maintenance and assembly.
1. Accessibility for ease of part repair and replacement.
2. Design for ease in assembly and disassembly.
3. Provide drawings for maintenance and method of assembly.

Process Engineering Functions

The process engineer must receive certain information from product engineering before he may proceed with his task. Information received is as follows:

Information from Product Engineering to Process Engineering—
A. Part print
B. Engineering release
    1. Production rate per hour, month, and year.
    2. Subassembly and assembly that part is a component of.
    3. Release date—date on which processing, tooling, and planning may proceed with work.
    4. Part name, number, and material.
C. Changed part print when revisions are made after the original print is distributed.
D. Engineering change release—similar to the engineering release when revisions are made on part, production, material, and names.

Functions of the process engineer are as follows:
A. To determine the order or sequence of operations necessary to manufacturing the part.
    1. Operation lineup or routing.
    2. Operation sketches.

B. To determine and order the tooling and gages necessary to manufacture the part.
    1. Orders to design.
    2. Orders to build.
    3. Orders to buy.
C. To determine, select, and order the equipment necessary to manufacture the part.
D. To determine the need for and originate orders for all revisions necessary when part print changes occur.
E. To follow up the tooling and equipment to determine if all is functioning as planned. If not, make the necessary revisions.

The functions and role of process engineering are graphically portrayed in Figure 182; the flow of information and responsibilities are illustrated.

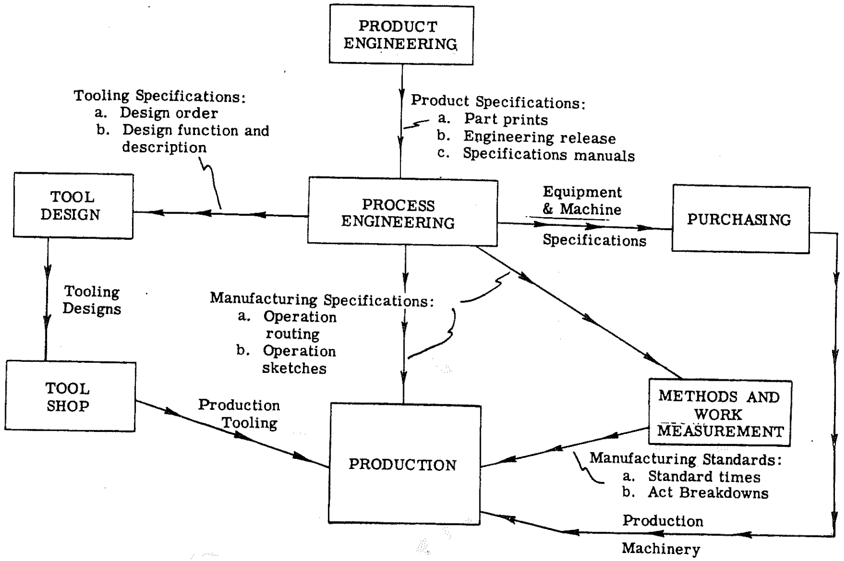


Figure 182. Role of Processing Engineering

This general relationship of departments exists in both large and small plants, regardless of the industry involved. In certain industries the functions are more distinct than they are in others.

Critical Dimensions

From the part print, the process engineer determines the "critical" dimensions. Critical dimensions apply to those areas of the part which must be held to the closest tolerances, have the best surface finish, or are the baseline of dimensioning. The process engineer must know which are the critical dimensions so that he can correctly determine the sequence of operations, equipment, and tooling necessary and the points of location to be used. Critical dimensions often provide arithmetic, geometric, or mechanical control of the workpiece.

Closer tolerances require that more precise tools and equipment be used, thus raising the product cost. Certain dimensions must be held to close tolerances to permit proper functioning of the part, accuracy after assembly, and controlled fits between parts. Several methods may be used

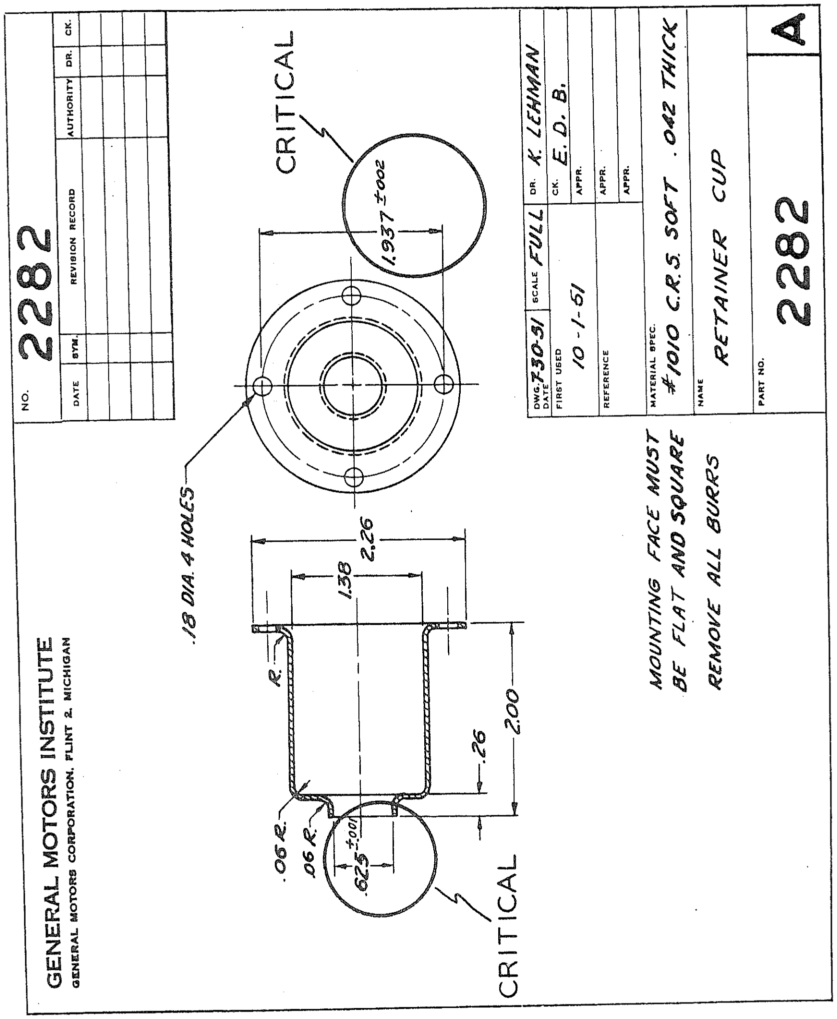


Figure 183. Close Tolerances

to identify critical dimensions, as follows:

1. Close tolerances (+ or – .005 or less would generally be critical for sheet metal).
2. Baseline of dimensioning.
3. Dimensions requiring close control of flatness, concentricity, roundness, squareness, and surface finish.
4. Dimensions that provide geometric control of the part.
5. Dimensions that provide mechanical control of the part.

Close tolerances. Close tolerances on a dimension denote a critical dimension. Closer tolerances are more difficult to obtain and require more expensive tooling and equipment. Also, rejection of parts is normally caused by poor control of critical dimensions. When determining the sequence of operations required to make the part, the process engineer must give priority to the critical dimensions.

Two dimensions of close tolerances are shown in Figure 183. The inside diameter of the flanged hole is held to close tolerances. Also, the centerline diameter, which locates the four holes punched in the flange, has close tolerances. The inside diameter of the drawn cup shown in Figure 184 has close tolerances. These are examples of close tolerances of drawn parts.

The diameters of all seven holes in the formed part shown in Figure 185 have close tolerances. The location of the holes is not critical. The flat part shown in Figure 186 has several dimensions that have close tolerances. The slotted and punched holes are critical, as well as the angles and some of the radii.

Baseline. The surface from which most dimensions on the part are taken is called the baseline of dimensioning. Actually, a part may have more than one baseline of dimensioning. The baseline is the reference point for the majority of dimensions. In some cases, this baseline may be the centerlines of a hole or cylinder.

On the drawn part shown in Figure 184, the top surface of the flange is a baseline of dimensions. On the flat part in Figure 186, the punched hole is one baseline of dimensioning. On the formed part in Figure 185, the centerlines of the part are the baselines.

The baseline of dimensioning is a critical area of a part. Since most dimensions are taken from the baseline, an error on the baseline would cause all dimensions to it to be in error.

Special dimensions. Special dimensions are often placed on parts to control flatness, concentricity, squareness, roundness, and surface finish. These dimensions are of the critical type. More care in production is necessary to control these dimensions, and often special tooling and equipment are required. Such dimensions are hard to maintain when

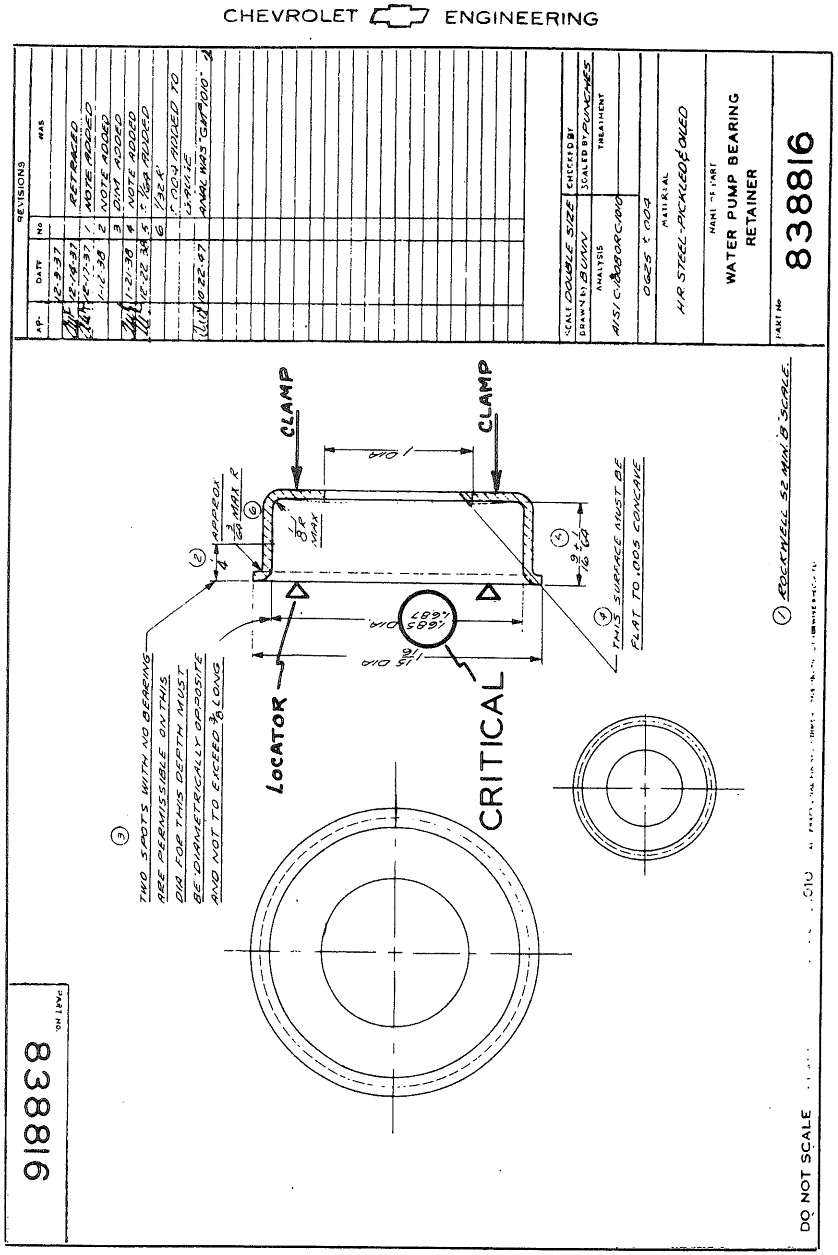


Figure 184. Close Tolerances

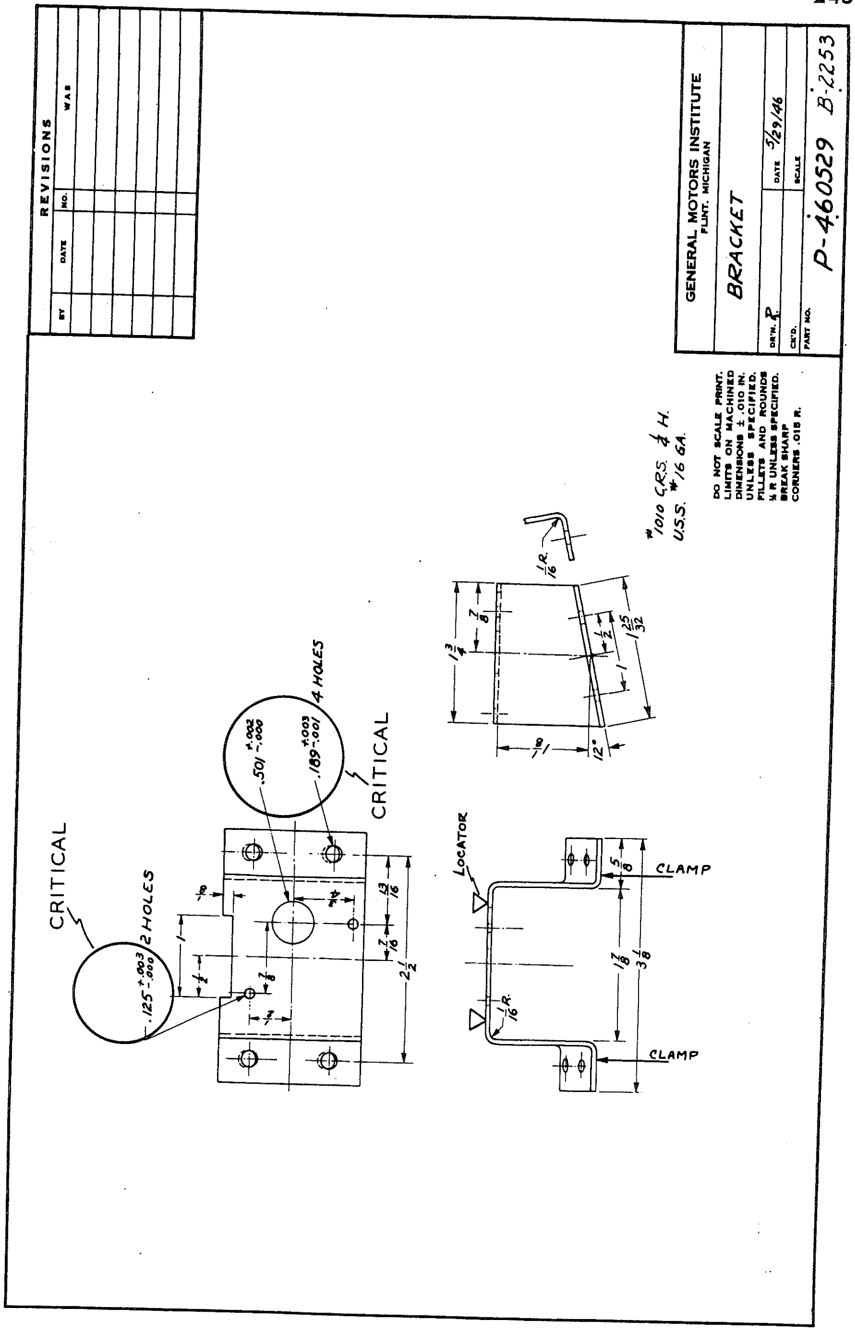


Figure 185. Close Tolerances

manufacturing sheet metal parts. The areas marked (1) and (2) on the flat part of Figure 186 are such critical dimensions. The hole must be square to the surface of the part over 80 per cent of the hole surface.

The note, "mounting face must be flat and square," on the drawn part of Figure 183 denotes a critical surface. The mounting surface is the top of the flange. In Figure 184, the note on the flatness of the cup bottom is a critical dimension.

Geometric control. The part must be located and held while further work is being performed on it. A surface on the part which provides a geometric shape against which locators may be positioned to control the part is called a critical surface. A comparison of several geometric shapes illustrates their preference as a surface for location:

Large flat surface (Plane)	Good
Small flat surface (Plane)	Poor
Several small flat surfaces (Planes)	Good
Cup, flanged hole (Cylinder)	Fair
Spherical cup bottom (Sphere)	Poor

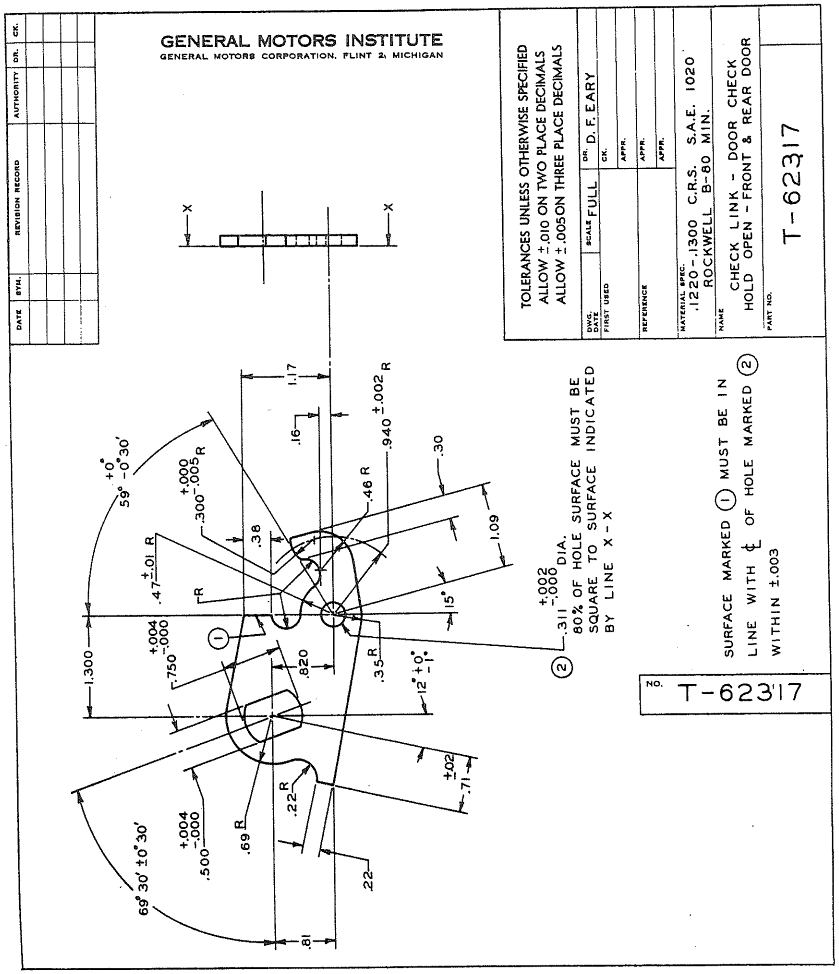

GENERAL MOTORS INSTITUTE
GENERAL MOTORS CORPORATION. FLINT 2, MICHIGAN
DATE
SYM.
REVISION RECORD
AUTHORITY
DR.
CK.
TOLERANCES UNLESS OTHERWISE SPECIFIED
ALLOW ±.010 ON TWO PLACE DECIMALS
ALLOW ±.005 ON THREE PLACE DECIMALS
SCALE FULL
DR. D. F. EARY
CK.
APPR.
APPR.
DWG. DATE
FIRST USED
REFERENCE
MATERIAL SPEC. .1220-.1300 C.R.S. S.A.E. 1020
ROCKWELL B-80 MIN.
NAME CHECK LINK - DOOR CHECK
HOLD OPEN - FRONT & REAR DOOR
PART NO. T-62317
X
X
1.17
.16
.940 ±.002 R
.30
.46 R
1.09
15°
.38
59° +0° -0°30'
.300 +.000 -.005 R
.47 ±.01 R
R
①
.750 +.004 -.000
1.300
.820
.35 R
12° +0° -1°
②
.311 +.002 -.000 DIA.
80% OF HOLE SURFACE MUST BE SQUARE TO SURFACE INDICATED BY LINE X-X
SURFACE MARKED ① MUST BE IN LINE WITH ℄ OF HOLE MARKED ② WITHIN ±.003
NO. T-62317
.22 R
.71 ±.02
.69 R
.500 +.004 -.000
69°30' ±0°30'
.22
.81

Figure 186. Close Tolerances

A large flat surface on a part permits spreading the locators further apart. Better control results from widespread location. This type of surface is preferred over all others for location. Refer to the formed part in Figure 187. If the small flat flange were used to locate the part, poor geometric control would result. The larger flat surface of the clip would be preferred.

In the case of a long cylindrical cup, the cylindrical surface provides better geometric control than does the small flat bottom. The locators can be spread further apart, thus providing better control. The larger geometric surface provides better control than does the smaller surface.

Mechanical control. A surface of the part against which clamping forces may be applied without distortion or marring of the part is called a critical surface. For example, refer to the drawn cup in Figure 184. If the locators were placed on the top surface of the flange, and clamps placed on the cup bottom, the bottom would be distorted past the allowances of the flatness note. In this case, the cup bottom does not provide mechanical control of the part. A preferred surface for clamping would be the back side of the flange. No part distortion would occur. Also notice that to be a critical surface by the mechanical control factor, the entire part must be rigidly held when clamped on that surface; or, at least, all portions must be sufficiently rigid for the work that is to be performed.

Normally, a surface with a high surface finish control would not be used for clamping. Marring of the surface may cause rejection of the part. Use of clamps covered with rubber or soft metal may be permissible.

Refer to the formed part in Figure 185. If locators were placed on the large flat top surface of the bracket and the clamps were placed on the two small flanges, distortion would result. As the clamping force is applied, the flanges would bend rather than clamp the part securely against the locators. The flanges offer poor mechanical control of the part. The clamps should be on the surface, just opposite the locators.

Types of Operations

All operations required to manufacture sheet metal parts fall into

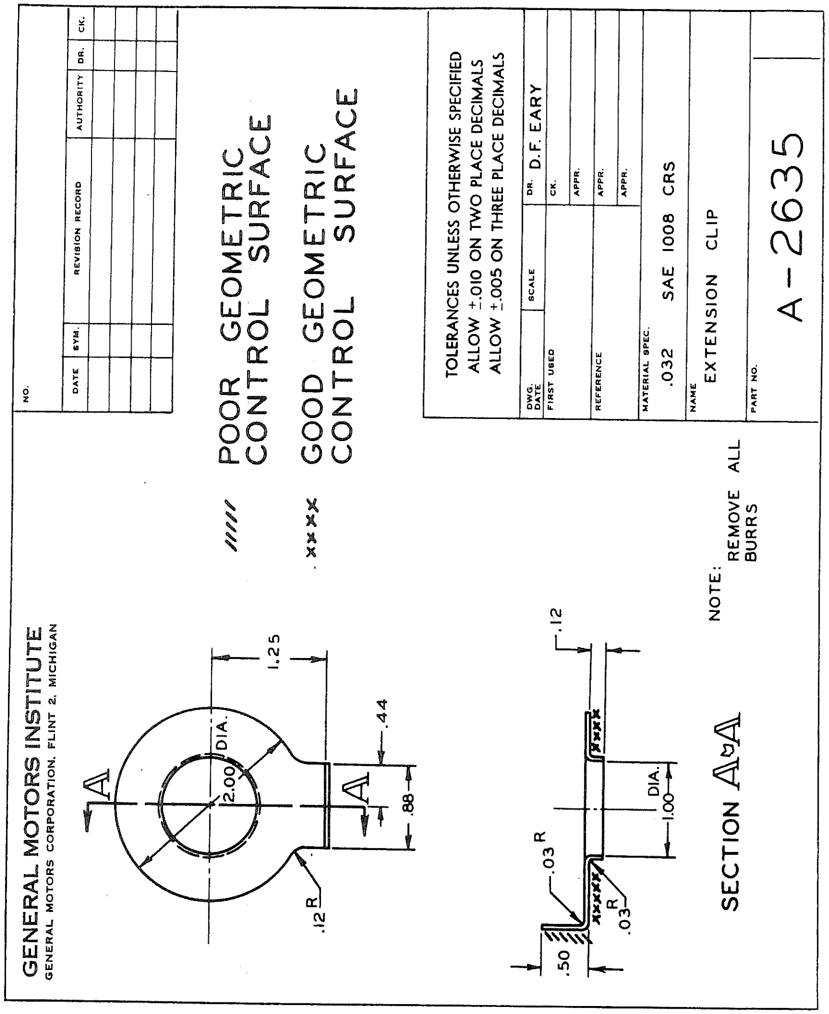

GENERAL MOTORS INSTITUTE
GENERAL MOTORS CORPORATION, FLINT 2, MICHIGAN
NO.
DATE
SYM.
REVISION RECORD
AUTHORITY
DR.
CK.
//// POOR GEOMETRIC CONTROL SURFACE
xxxx GOOD GEOMETRIC CONTROL SURFACE
TOLERANCES UNLESS OTHERWISE SPECIFIED
ALLOW ±.010 ON TWO PLACE DECIMALS
ALLOW ±.005 ON THREE PLACE DECIMALS
SCALE
DR. D.F. EARY
CK.
APPR.
APPR.
APPR.
DWG. DATE
FIRST USED
REFERENCE
MATERIAL SPEC. .032 SAE 1008 CRS
NAME EXTENSION CLIP
PART NO. A-2635
NOTE: REMOVE ALL BURRS
1.25
2.00 DIA.
.44
.88
.12 R
A
A
.12
.03 R
1.00 DIA.
.03 R
.50
SECTION A-A

Figure 187. Geometric Control

several categories. These categories are groupings of operations by their function or purpose. On one part, for example, the operation may fall into the critical operation category. On a different part, the same operation may fall into another category. Therefore, the categories are grouped by purpose and not by the operation description. These operation categories are as follows:

1. Sheet Metal Working Operations
   a. Critical

   b. Non-critical
   c. Tie-in
2. Allied Operations
   a. Cleaning
   b. Surface treatment
   c. Surface coatings
   d. Heat treatment
   e. Machining
3. Inspection Operations
   a. Receiving
   b. Production
   c. Final

Assembly of sheet metal parts is not discussed in this text; therefore, operations used in assembly and processing for assembly are not included in this chapter.

Critical operations. Critical operations are those used to produce or obtain critical dimensions or surfaces. The critical dimensions and surfaces have already been defined. Due to the precision required, critical operations receive top priority when determining the operational sequence. If punching is used to cut a hole to plus or minus one-thousandth of an inch in diameter, the operation is critical. If the hole were to be cut to plus or minus ten-thousandths of an inch in diameter, the operation would not be critical. Critical operations are determined by the closeness of tolerance required and not by the operation description, such as punching, blanking, or trimming. Critical operations are also used to establish surfaces that are a baseline of dimensioning, that are good for geometric control, or that allow mechanical control of the part.

Non-critical operations. Even though an operation is non-critical, it is still vital for production of the part. Non-critical means that the tolerances on the dimension produced are large and that difficulty is not expected in controlling the operation. When tolerances of plus or minus ten thousandths of an inch are involved, the operation is usually non-critical. This is also true for greater tolerances. Greater tool wear is allowed before non-critical dimensions are thrown out of tolerance. Usually, non-critical operations are lower in cost. The ideal situation would be to have only non-critical operations, for cost would be at a minimum. Critical operations are required to achieve function and assembly characteristics.

Tie-in operations. The sheet metal working operations may also be called tie-in operations when they perform a distinct function. Often, after forming or drawing, sheet metal parts may be heat treated. The critical and non-critical dimensions previously produced go out of tolerance during heat treatment. An operation is now necessary to bring the part back to shape before further working. This operation of returning the part to shape is called the tie-in operation.

When large sheet metal parts are made, the shape and contour are first established at the draw operation. The parts are still somewhat flimsy after drawing. During subsequent trim, punch, and flange operations, the part becomes distorted due to handling and loading and unloading from the die. A tie-in operation is necessary to bring the contour of the part back to its original drawn shape. This tie-in operation is often called the restrike or spank operation. If the surface being brought back to shape is flat, the operation is called flattening.

Tie-in operations may be required after any operation that tends to distort or warp the part. Such operations might include tumbling, sandblasting, assembly by welding, deburring, and cleaning. The tie-in operation may be required after a sheet metal working operation or an allied operation.

Allied operations. All operations performed on the sheet metal part which are not inspection or sheet metal working operations are classified as allied operations. The sheet metal working operations were defined under cutting, forming, and drawing. Allied operations are those operations that are not performed with a die in a press. Allied operations are not stamping operations. Checking or inspection of parts with gages are operations having their own category.

Allied operations commonly used on sheet metal parts include:

1. Cleaning Operations
   a. Washing
   b. Degreasing
   c. Pickling
   d. Sand Blast
   e. Steam
   f. Hydro-Blast
   g. Shot Blast
   h. Tumbling

2. Surface Treatment Operations
   a. Buffing
   b. Polishing
   c. Vapor-blast
   d. Sand-blast
   e. Tumbling

3. Surface Coating Operations
   a. Anodize
   b. Plate
   c. Paint
   d. Bonderize
   e. Parkerize
   f. Rust preventative dip
   g. Metallize
   h. Flame plate

4. Heat Treatment Operations
   a. Anneal
   b. Stress Relieve
   c. Carburize
   d. Nitride

   e. Normalize
   f. Harden
   g. Cyanide

5. Machining Operations
   a. Reaming
   b. Tapping
   c. Broaching
   d. Milling

Inspection operations. Inspection operations fall into three main groups. First, the raw material purchased from other plants must be inspected to see that all specifications are met. Second, the parts may be inspected between production operations. Inspection at the production operation usually involves critical dimensions or surfaces. The cost of inspection is necessary to find quickly any variations of critical dimensions. Due to close tolerances, critical dimensions are more likely to fall out of limits. The cost of inspection at the production operation is less than the cost of repairing or scrapping parts rejected at final inspection.

After all production operations are finished and the part is complete to all part print specifications, another inspection operation is necessary. This operation is called final inspection. The parts are inspected completely. The parts are inspected by a department separate from the production department. Dimensions not inspected by production gages, as well as those already inspected, are reinspected at final inspection.

Inspection operations on sheet-metal parts and raw materials include the following:

1. Receiving Inspection
   a. Sheet metal thickness and width
   b. Ductility
   c. Drawability
   d. Tensile strength
   e. Springback
   f. Hardness
   g. Shear strength
   h. Surface finish

2. Production and Final Inspection
   a. Plug gages
   b. Templates
   c. Ring gages
   d. Feeler gages
   e. Contour gages
   f. Thickness of plating gages
   g. Surface defects and scratches
   h. Magnetic particle test
   i. Fluorescent penetrant test
   j. X-ray

- k. Snap gages
- l. Ultra-sonic test
- m. Dial indicator gages
- n. Optical comparators

Many other factors are inspected depending on part print specifications. The majority of operations have been shown.

Operations Required

After determining which are the critical dimensions on the part print, the operations required for making the part should be determined. For flat and formed parts, the operations needed can be determined by observing the part print. For drawn parts (especially cups) the blank diameter, percent reductions, and cup diameters must be calculated first. Then the number of redraws required, if any, can be found. These calculations have already been described in detail. Such calculations are necessary before further processing can be accomplished.

Refer to Figure 186. The operations required to make the part might include:

1. Blank
2. Punch
3. Slot
4. Shave (Punched hole)
5. Shave (Surface marked by number one)

The allied and inspection operations needed are:

1. Receive inspect (sheet metal)
2. Production inspect
3. Wash
4. Final inspect

The operations required to make the formed part shown in Figure 185 may include:

1. Blank
2. Bend (2 places)
3. Flange (2 places)
4. Punch (7 holes)

Other operations needed are:

1. Receive inspect (sheet metal)
2. Production inspect
3. Wash
4. Paint
5. Final inspect

The operations required to make the drawn part shown in Figure 183 may include:

1. Blank
2. Flange hole

3. Draw
4. Redraw
5. Redraw
6. Punch hole in bottom
7. Punch (4) holes in flange
8. Trim
9. Restrike flange to flatten

Other operations necessary are:

1. Receive inspect (sheet metal)
2. Production inspect
3. Degrease
4. Chrome plate
5. Final inspect

To be able to determine the operations required, the processor must have a knowledge of the terminology used to identify sheet metal operations and must know what the purpose and limitations of each operation are. When allied operations are required, it is best that the process engineer seek the advice of specialists in the field concerned. The metallurgist, chemist, and other trained personnel have a greater knowledge in the areas of heat treatment, plating, painting, and other such operations. The processor is not expected to achieve perfection in all fields. He is expected to be the investigator and coordinator in obtaining necessary information from other sources.

Sequence of Operations

The sequence of operations may be varied in some cases; at other times, the sequence is limited. For instance, you would not draw a part and then blank it. Likewise, you would not trim a part and then draw it. Certain operations must follow others.

For some parts, more than one sequence may be used successfully to manufacture the parts. Such liberal variation in operation sequence is not always practical or possible, however. The cost of producing the part may not be equal for all sequences, and the economy of the sequence is an important factor. At other times, the specifications on the part print require that a certain sequence be used. For example, when the location of a hole is critical and that hole is in the flange, the hole is usually punched after the flanging operation. Actually, it would be less expensive from a tooling standpoint to punch the hole while the part was flat, but it is very difficult to maintain hole location with such a sequence. Several rules to follow in determining operation sequence are:

- A. Punching, Slotting, and Perforating
  1. Holes near the edge of the part should be cut after blanking. The cutting action during blanking might distort the hole.
  2. When large and small holes are located close to each other, the large hole should be cut first. The cutting action during cutting of the large hole might distort the small hole.

3. When a hole is located near the radius of a formed part, the hole should be cut after forming. Forming might distort the hole.
4. When a hole is located in an area that is wiped, burnished, or ironed by the forming steels, the hole should be cut after forming to prevent distortion of the hole.
5. Holes in drawn parts must be cut after drawing. Holes in high stress areas would cause failure of the metal. Also, the holes would be distorted severely.
6. Whenever possible, holes should be cut when the part is flat. After forming or drawing, expensive cams might be required or the part might have to be tipped to an awkward angle. In either case, die cost would be higher.
7. Holes should be cut after forming or drawing when close tolerances on hole location must be maintained. Predicting hole location before forming or drawing would give only an approximate location.
8. When high hole flanges are to be formed, reaming of the punched hole will reduce breaking of the flange. Removal of the fractured edge reduces stress concentrations. Shaving of the hole will also aid in the same manner.
9. When several holes are cut in the flange or bottom of a cup, they should be cut simultaneously, if possible. Otherwise a second die with gages in the first holes would be necessary, and maintenance of close tolerances on hole location would be difficult.
10. When pilots enter a hole for location of the progressive strip, the hole should be cut undersize if the diameter is critical. The hole is then shaved to size at a final station.

B. Trimming
   1. Drawn parts should be trimmed before flanging or hemming. Trimming after hemming would be impossible. Trimming after flanging would require expensive cams in the die in many cases.
   2. It is often necessary to trim large irregular-drawn parts in more than one die. To trim the entire part at once would require cams. Partial trimming permits tipping of the panel to eliminate the need for cams.

C. Drawing, flanging, hemming, and bending
   1. Drawn parts must be flanged after the drawing operation. Flanging during the drawing operation would prevent metal flow and wrinkles would occur in the flange because no

blankholding pressure could be applied there. The same is true for hemming.
   2. The part edge is first flanged and then the flange is formed over further to obtain the hem. To form a hem in one operation generally requires the use of expensive cams.
   3. It is generally more efficient to flange first and bend the part afterwards.

The limitations listed control the sequence of operations for many sheet metal parts. Other more special limitations may occur, but those listed are the most common.

The operations required and their sequence for the drawn cup in Figure 184 are shown below. The operations are identified as to type and the reason for the classification is given.

Operation	*Type*	*Reason*
Receiving Inspection	Inspection	To check quality of incoming sheet metal
Roller Level	Allied	To equalize strength
Blank	Non-critical	Large tolerance allowed
Draw	Critical	Origination of geometric shape
Wash	Allied	Remove drawing lubricant
Punch	Non-critical	Large tolerance allowed
Trim	Non-critical	Large tolerance allowed
Restrike Cup Bottom	Critical	Close tolerance on flatness
Production Inspection	Inspection	For critical dimension
Size Cup Inside Diameter	Critical	Close tolerance on diameter
Production Inspection	Inspection	For critical dimension
Wash	Allied	Remove lubricants
Final Inspection	Inspection	All dimensions checked
Rust Preventative Dip	Allied	Preparation for shipping

No tie-in operations are included. Since no heat treatment was used and the part was sufficiently rigid to prevent distortion due to handling,

80540

the tie-in operation was not required. Examples of the other types of operations are illustrated. Note that the origination of the cup or geometric shape is classified as a critical operation. The top flange surface is the baseline of dimensioning in one direction. The cup centerline is the other baseline. Both baselines are originated during the draw operation. Also, the flange represents the best geometric surface for locating the part. The cup bottom and side wall cylinder are second best for location. All of these surfaces are originated in the draw die. For these reasons, drawing is a critical operation.

In the sequence of operations, the critical operations involving flatness and close tolerances are completed last. If performed first, these critical surfaces may become distorted during punching or trimming. Thus the critical operations are given top consideration when determining the sequence of operations.

The part shown is painted after assembly. Therefore, rust preventative is placed on the part prior to shipment. If the part was to be assembled immediately, this operation would not be necessary.

In many plants, the production inspection is not listed as a separate operation. In such a case, this inspection would be included in the critical operation.

Studies of operation sequences of any sheet metal part should reveal priority given to the critical operations. Also, each operation can be classified as to type, such as critical, non-critical, or allied.

An allied operation such as machining might be critical if the tolerances are close on the dimension in question.

Systems of Location

The process engineer must determine how the sheet metal part is to be located in the die, and the die designer must provide gages to accomplish this location. The die designer is responsible for the physical shape and style of gages, but he does not determine their location.

Several symbols are used to designate locators, clamps and supports as follows:

Triangle	△	Side View of a Locator
Triangle with Point at Center	◬	Top View of a Locator That Is Hidden
Triangle with Cross at Center	⨹	Top View of a Locator
Arrow	←	Clamp
Arrow Inside of Triangle	◁—	Clamp and Locator
Block with Letter "F"	$\boxed{F}$	Fixed Support
Block with Letter "A"	$\boxed{A}$	Adjustable Support

If a three-dimension model is being used, the locator appears as a pyramid and the support as a cube. Actually, these symbols are the pyramid and cube as they would appear in a two-dimensional drawing.

Six-point location. The number of locators needed to position a part or to control that part physically will now be discussed. If one locator is used, only a point of gaging has been obtained. If two locators are used, a line of gaging has been established. Three locators establish a plane of gaging. For many geometric shapes, six points of location are necessary to control the part.

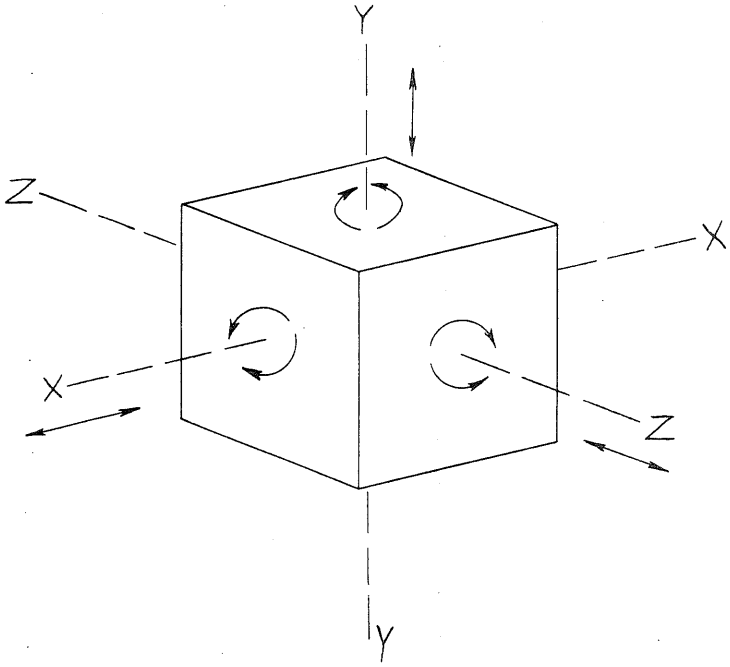
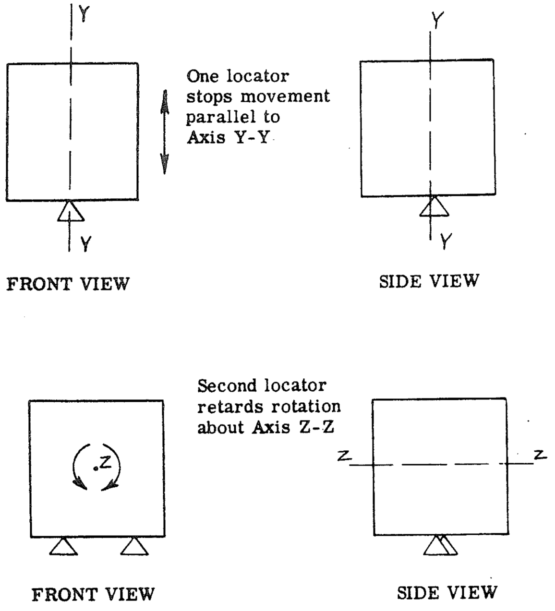


Figure 188. Movements of a Cube in Space

To understand the principle of part location, the movements of a part in space must be analyzed. Refer to Figure 188, which shows a cube. The axes of the cube are designated as *X–X*, *Y–Y*, and *Z–Z*. There are six basic movements of this cube in space. These movements are:

1. Movement parallel to Axis *X–X*.
2. Movement parallel to Axis *Y–Y*.
3. Movement parallel to Axis *Z–Z*.
4. Rotation about Axis *X–X*.
5. Rotation about Axis *Y–Y*.
6. Rotation about Axis *Z–Z*.

To locate the cube completely, all six movements must be retarded. If one locator were placed under the cube, only the movement parallel to axis *Y–Y* would be retarded. If a second locator were placed under the cube, the rotation about axis *Z–Z* would be retarded. These conditions are illustrated in Figure 189. Note that only the movement and rotation are retarded. A clamp is required to control the part completely in these two directions.



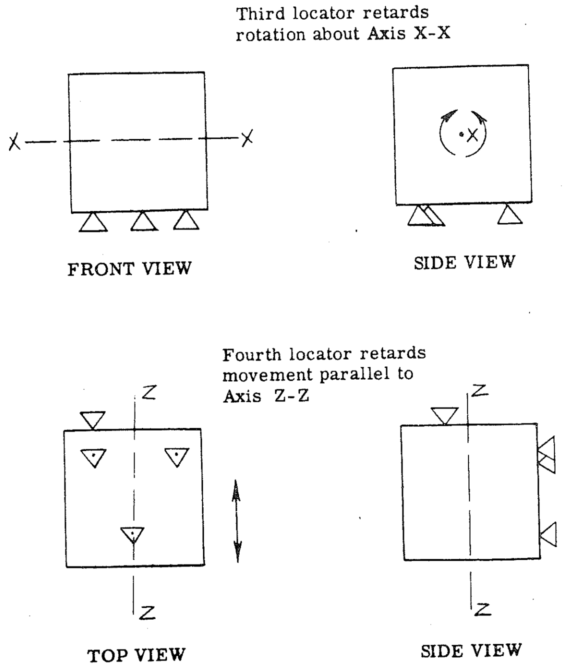
One locator stops movement parallel to Axis Y-Y

Second locator retards rotation about Axis Z-Z

Figure 189. One and Two Locators on a Cube

When a third locator is placed under the cube, the rotation about axis *X–X* is retarded. With three such locators on one surface, the position of a geometric plane has been determined. Thus, three points are required to determine a plane. At this time, the cube is still free to move in the remaining three directions.

A fourth locator has been placed on another surface of the cube, as shown in Figure 190. The movement parallel to axis *Z–Z* has been retarded. Another locator on the same surface retards rotation about axis *Y–Y*. These two locators on the same surface establish a line of gaging. Thus, two points determine a line.



Third locator retards rotation about Axis X-X

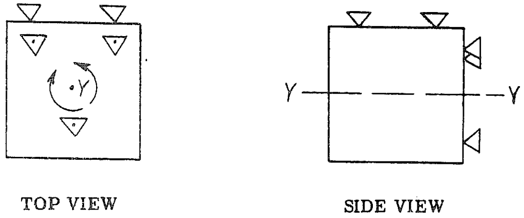
Fourth locator retards movement parallel to Axis Z-Z

Figure 190. Three and Four Locators on a Cube

The sixth locator is placed on a new surface, as shown in Figure 191. The movement parallel to axis *X–X* is now retarded. When clamped against the six locators, the cube is completely controlled. Notice that locators are not placed on a surface parallel to a surface that has other locators on it. When locators are placed on parallel surfaces, they do not restrict one movement per locator; an excess of locators exists.

Figures 192, 193, 194, and 195 illustrate other geometric shapes and the location required for them. A rectangular solid would require six locators, as did the cube. The pyramid needs six locators for control, as illustrated. A different situation occurs with the long cylinder. A rotation about the cylinder centerline cannot be retarded by a locator. Therefore, only five locators are used. To retard the rotation, friction must be

Fifth locator retards rotation about Axis Y-Y



Sixth locator retards movement parallel to Axis X-X

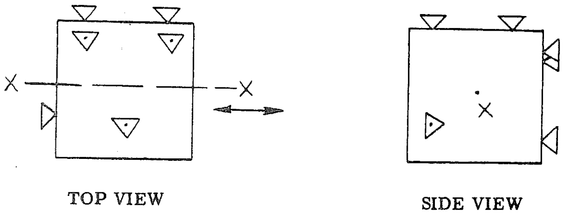


Figure 191. Five and Six Locators on a Cube

used. When a round bar is gripped in a chuck or collet, rotation is stopped by the force due to friction between the jaws and the bar. The force is increased by increasing the clamping force. In some chucks, rotation is stopped by serrated jaws, which must mutilate or distort the part.

Only three locators are used to control a sphere. Friction or part distortion must be used to retard the three rotational motions of the sphere. Whenever a part is distorted, a small plane or planes are produced to provide surfaces for location. In some cases, it is not permissible to distort the part.

Refer again to the long cylinder. The two pairs of locators resting on the cylindrical surface could represent the location provided by the conventional v-block. One pair of such locators would represent a v-block having a knife-edge. Two pairs of such locators would represent a long v-block. Refer now to the short cylinder shown in the illustration. This short cylinder might be a round sheet metal blank. Certainly this cylinder could not be located or controlled by a v-block. The surface or "plane" of the blank is positioned by three locators. Two locators on the blank edge restrict two motions. The sixth motion or rotation about the blank center cannot be retarded.

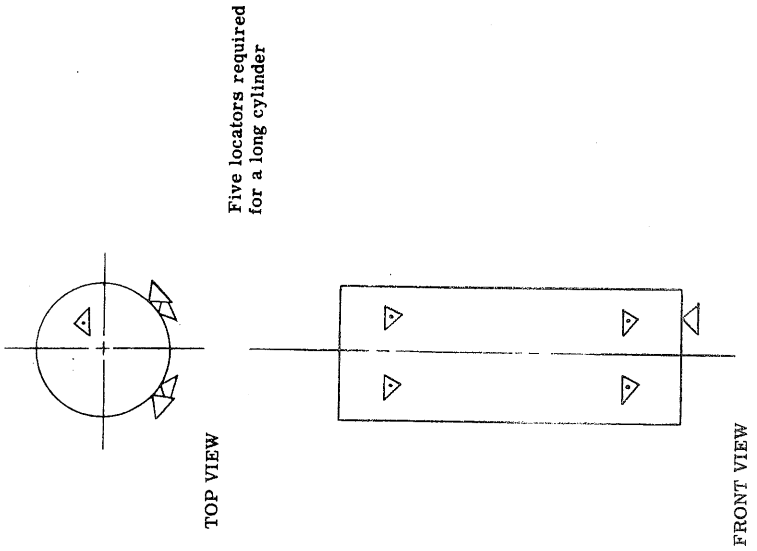


Figure 193. Location of a Long Cylinder

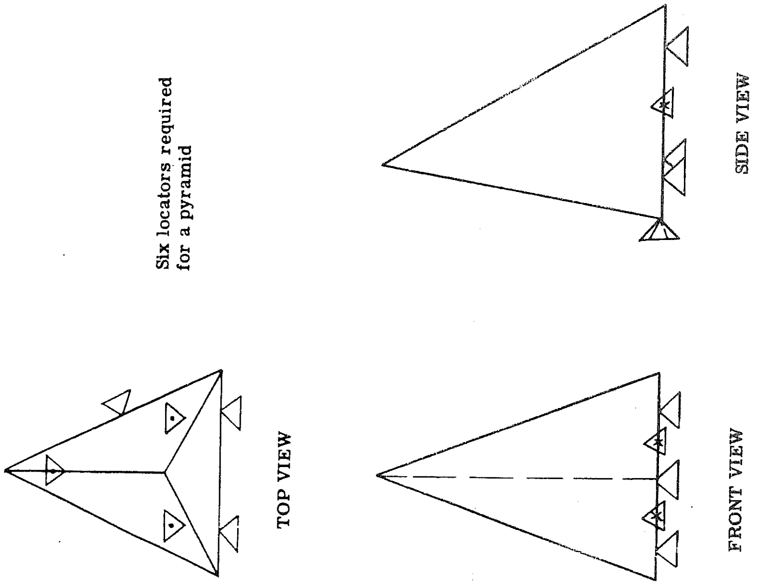


Figure 192. Location of a Pyramid

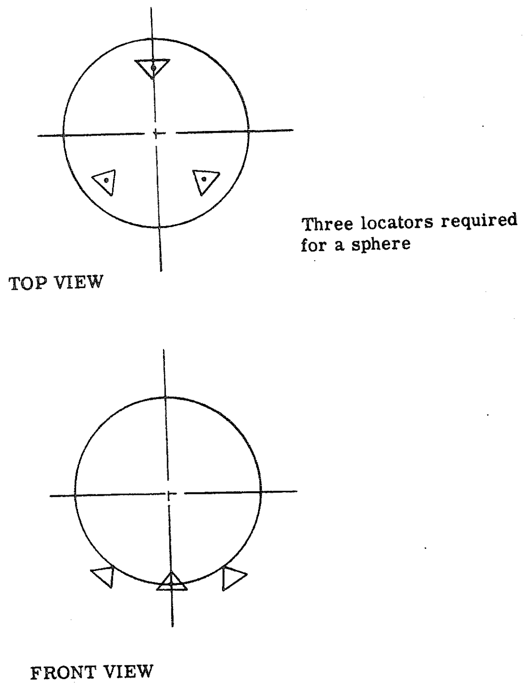


Figure 194. Location of a Sphere

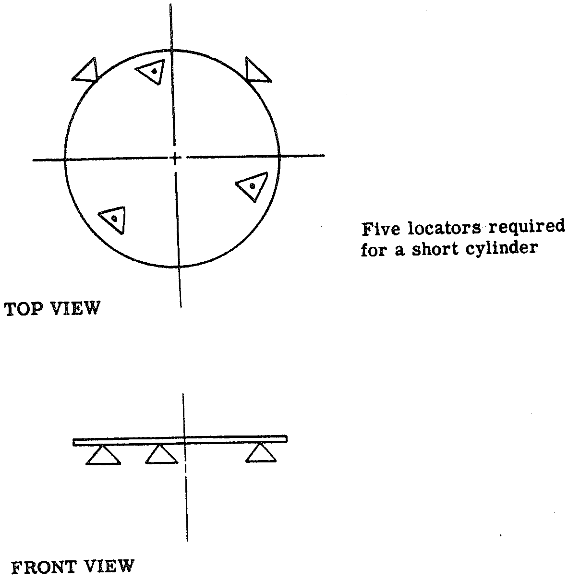


Figure 195. Location of a Short Cylinder

Another factor to consider in placing locators was illustrated by the long and short cylinders. This factor is geometric control. Geometric control was previously defined as a factor determining a critical dimension. Due to size, the long cylindrical surface was better for control than were the small flat ends of the cylinder. If three locators were placed on the end, the cylinder would not be rigidly located. The opposite was true for the short cylinder. The cylinder end is the largest geometric shape against which locators may be placed. This cylinder is more rigidly located by three points on the large flat surface. Thus the larger surface on a part provides better geometric control because the locators may be spaced further apart. For good geometric control, most of the locators must be on the larger surface.

Several rules now evolve, as follows:

Cubes

1. Since all surfaces are equal in size, no priority for location.

Rectangular Solids

1. Three locators for determining a plane should be on the largest surface.
2. Two locators for determining a line should be on the second largest surface.
3. One locator should be on the smallest surface.

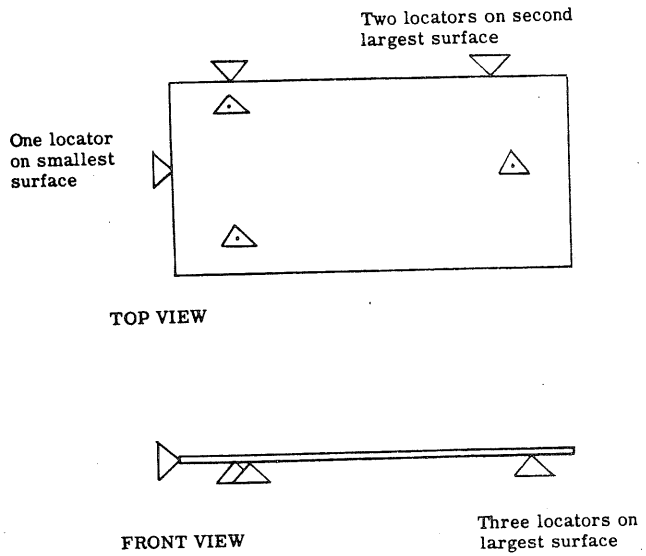


Figure 196. Location of Rectangular Sheet Metal

Cylindrical Solids

1. Long cylinders—four points on cylindrical surface and one point on end.
2. Short cylinders—three points on end and two points on cylindrical surface.

If a rectangular piece of sheet metal is to be located, position of locators should be as shown in Figure 196. Locators should be as far apart as possible.

Location from holes. If a hole is a critical dimension on a part, it may be desirable to locate from it. When the dimension being obtained is taken from the hole centerline, the close tolerances on the hole make it suitable for location.

Either one or two locators may be placed in a hole. One locator would retard one movement parallel to an axis. Two locators would retard two movements in a parallel direction. Rotation of a part cannot be retarded

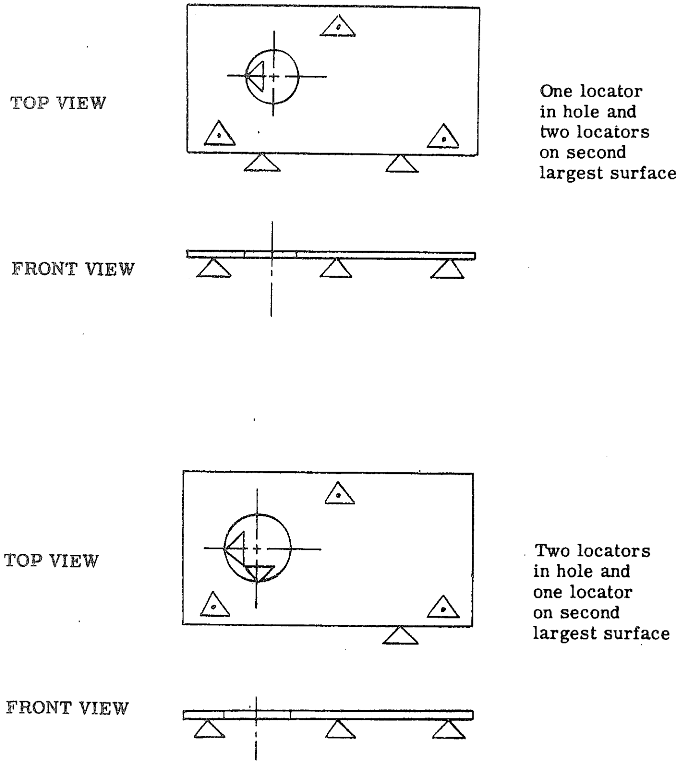

TOP VIEW
One locator in hole and two locators on second largest surface
FRONT VIEW
TOP VIEW
Two locators in hole and one locator on second largest surface
FRONT VIEW

Figure 197. **Locators with One Hole in Part**

by locators in a single hole; location in two holes is necessary to retard rotation. Figures 197 and 198 illustrate the use of locators in holes under several conditions. A single locator in a hole is accomplished with a diamond pin. A round pin affords two points of location from a hole.

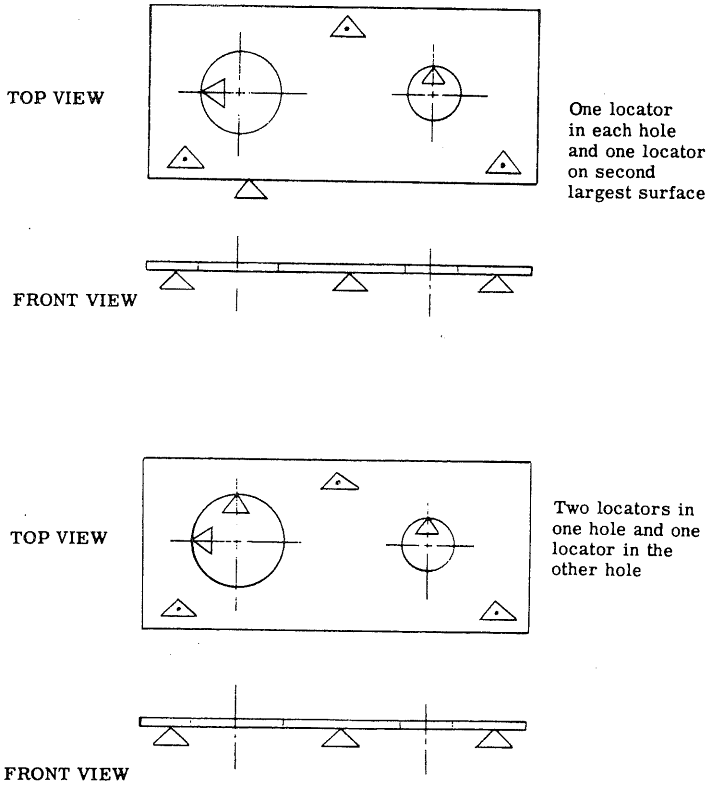
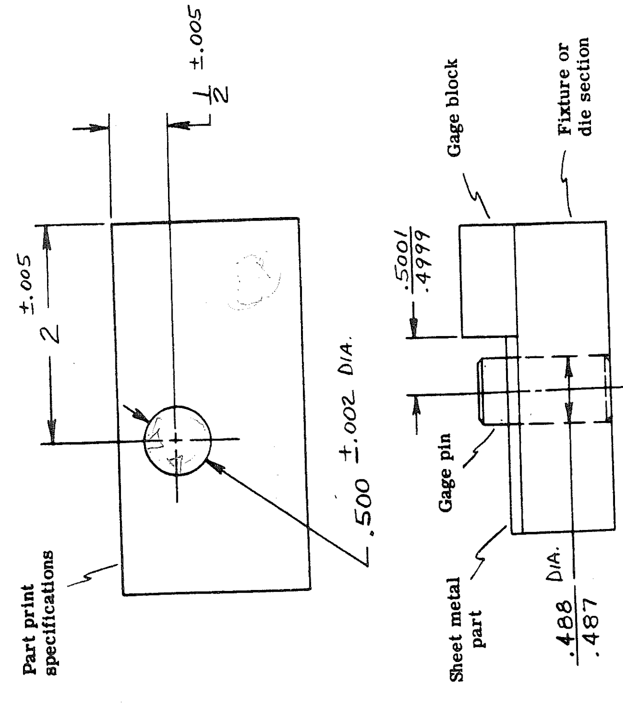

TOP VIEW
One locator in each hole and one locator on second largest surface
FRONT VIEW
TOP VIEW
Two locators in one hole and one locator in the other hole
FRONT VIEW

Figure 198. **Locators with Two Holes in Part**

If a hole is placed over a tapered pin, two points of location are obtained. The tapered pin may be spring-loaded to adjust to variations of hole diameter. An expanding three-section mandrel placed in a hole affords two points of location. The tapered pin and expanding mandrel offer positive location. The solid and diamond pins allow some part shift because the pin must be undersize to allow for hole tolerances on both diameter and location.

Dimensioning of pins that are used for locators is illustrated in Figures 199, 200, and 201. Notice that tolerances on hole diameter and centerdistance must be allowed for on the round pin. Only tolerances on hole diameter must be allowed for on the diamond pin. The diamond



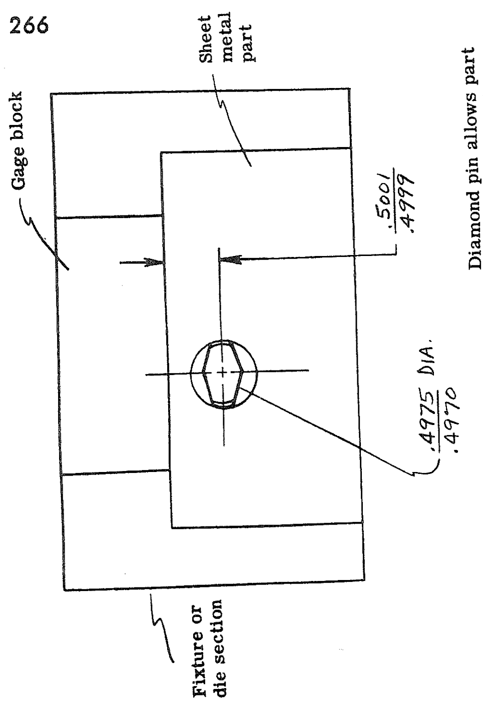
ROUND GAGE PIN

Pin diameter = Nominal hole diameter - Tolerance on diameter - Tolerances on hole location

D = .500 - .002 - .010

D = .488 inches

Figure 199. Dimensioning of Round Pin



Diamond pin allows part to shift so that part is located against the gage block. Therefore hole location tolerances do not have to be subtracted to find the pin diameter.

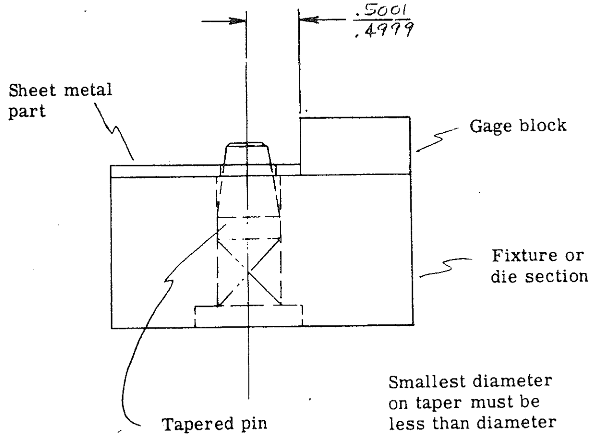
DIAMOND GAGE PIN

Pin diameter = Nominal hole diameter - Tolerance on diameter - Allowance for clearance

D = .500 - .002 - .0005

D = .4975 inches

Figure 200. Dimensioning of Diamond Pin



Smallest diameter on taper must be less than diameter of the round pin. Largest diameter on taper must be more than largest hole diameter.

Error expected with various pins:

Round pin	.007 inches maximum
Diamond pin	.0022 inches maximum
Tapered pin	.000 inches

Figure 201. Dimensioning of Tapered Pin

shape allows shift of the hole in one direction but maintains location in the other direction.

Supports. The ideal situation is to have clamps just opposite the locators. Deflection or distortion of the part is thus prevented. Often, however, due to the part shape, the locators are widespread for geometric control and the clamps placed away from the locators. Severe deflection occurs as shown in Figure 202. To limit deflection, a fixed support is placed at the point of highest deflection. The support is slightly lower than the locators. The locators still position the part, and the support limits deflection.

If better control of deflection is desired, the more expensive adjustable support is used. The part is placed on the locators. The support is adjusted upward until it just contacts the part, and the clamping force is then applied. The deflection is practically eliminated, but cost is increased.

Alternate locators. In most dies, it is not practical to clamp the part against the locators. The part may be clamped by the stripper plate, pad, or blankholder, but the part could shift out of position before being clamped. Therefore, alternate locators or retainers are utilized. Alternate locators for several sheet metal parts are shown in Figure 203. It is often

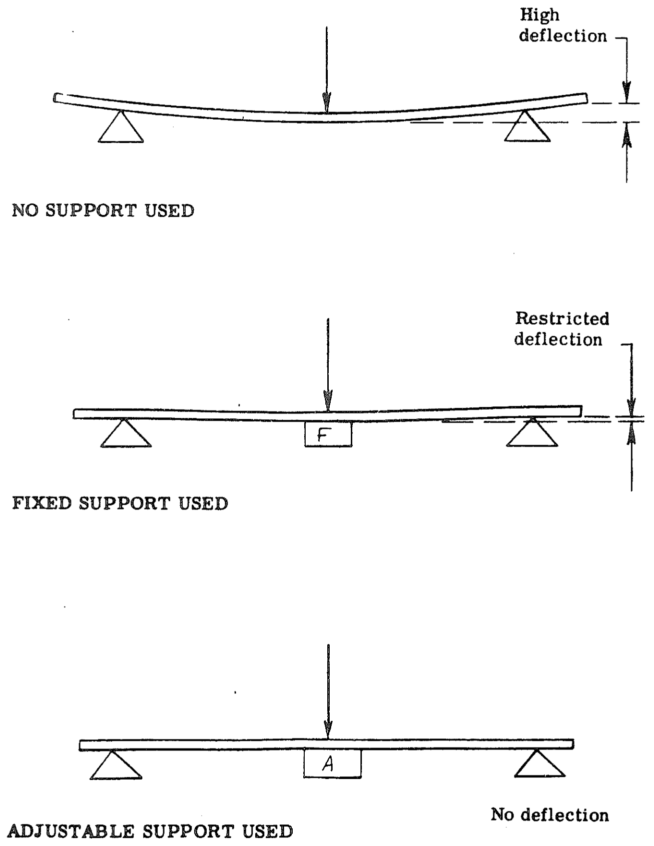


Figure 202. Use of Supports

difficult to distinguish which of the locators is the alternate locator. Whenever more than six locators are being used, it is likely that alternate locators are present. If more than five locators are used with a cylindrical shape, the same is true.

The nest used to locate parts in a die actually uses alternate locators to help retain the part. The solid stripper channel is a good example of alternate locators in use.

In most dies, the top surface of the punch steel, die steel, stripper plate, pad, or blankholder is used as a plane of location. Thus a flat surface is used rather than three individual locators.

Care must be taken when using alternate locators. If the part locates from the alternate locator, the tolerances may go out of control. The part may not be resting against the desired locator. Use of alternate locators is limited to cases where tolerances are large.

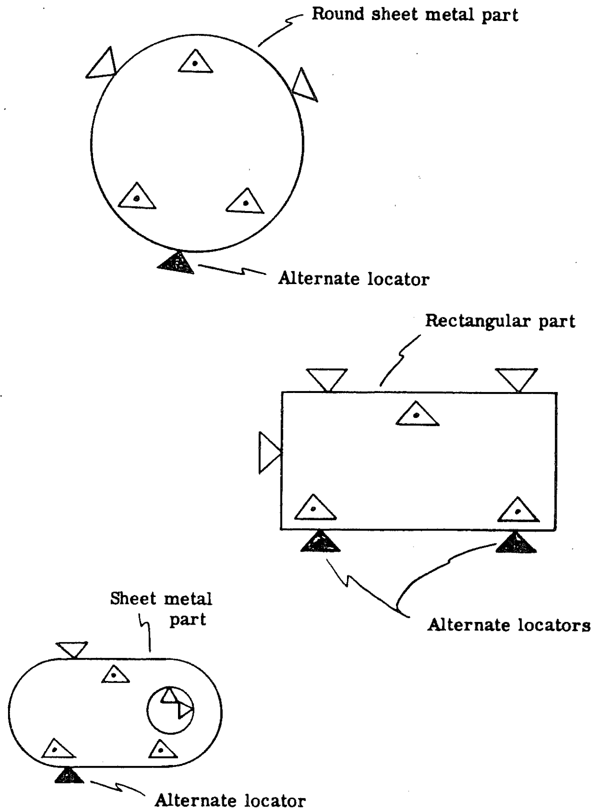


Figure 203. Alternate Locators

Often the drawn part is located by the punch steel or die steel when trimming or flanging. The center of the steel is machined to the part contour and provides a nest for location. This method of location is often the most efficient for irregular-shaped panels. Also, for large irregular panels, the center of the relatively flimsy panel is supported by the nest. This type of nesting is costly, however, because of the contour machining involved.

The system of location described may be used to select points of location for parts having a combination of geometric shapes.

Tolerance stacking. Considerable difficulty may be encountered when care is not taken in placing locators. General rules to follow are:

1. The inspection gages should locate the part at the same points as did the die.
2. The locators in the die should be on the surface from which the desired dimension being obtained is shown on the part print.

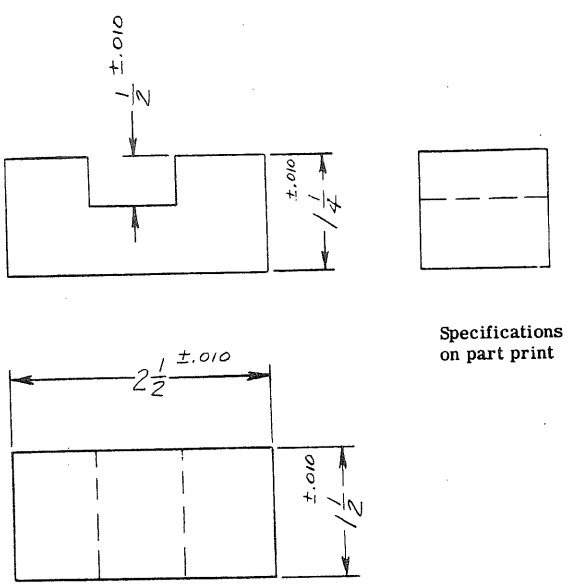


Figure 204. Specifications of Part Print

If either rule is violated, tolerance stacking will result. The dimensions on a part are shown in Figure 204. The sequence of operations and locating points used to make the part are shown in Figure 205. The tolerances have been violated on the final dimension. The tolerances on the final dimension could have been met by reducing the tolerances on the previous dimensions. This reduction of tolerances below those on the part print would increase cost, for more regrinds and tool adjustments would be required.

By changing the position of the locators, as shown in Figure 206, all dimensions can be held within the part print tolerances. Reduction of tolerance and increased cost are not necessary. Inspection gaging could involve tolerance stacking in the same manner if gage and fixture location points are not identical. The situation in a die is almost identical to that occurring in the fixture and gage.

Operation Sketches

Operation sketches are usually made by or controlled by the process engineer. These sketches show the part as it actually appears at each operation. The part print shows only the final part shape. Generally, it is too difficult to process a part mentally, for, with the part print, processors might specify locators on a hole that has not yet been cut in the part. Reliance on memory often results in costly errors. Use of the operation

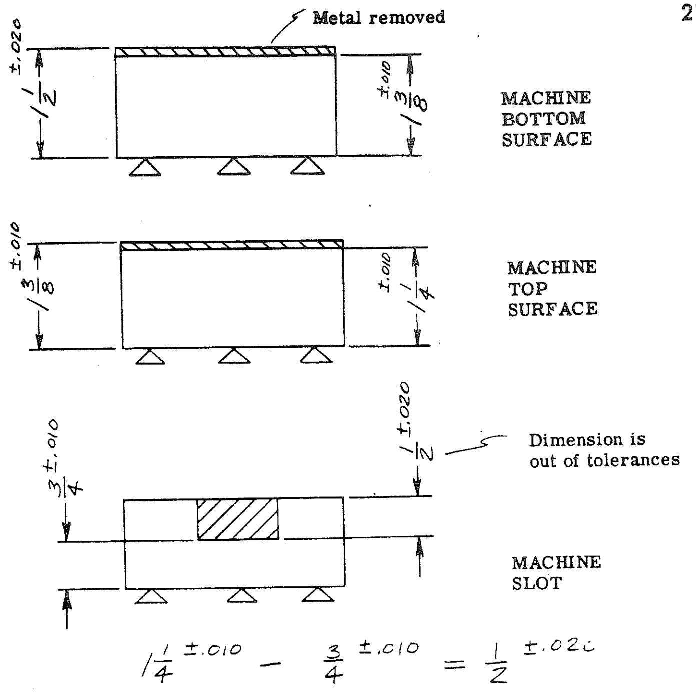
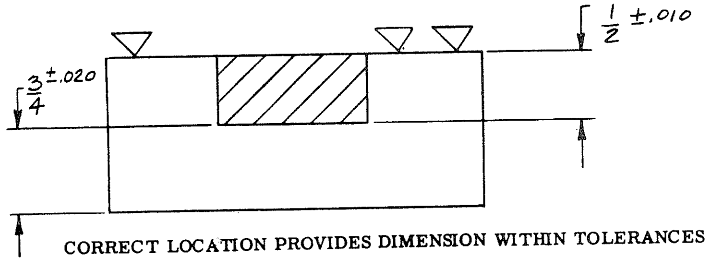


Figure 205. Tolerance Stacking



The part is located on the surface
from which the dimension is given
on the part print.

Alternate conditions:

1. The one-half inch dimension could have been held within the tolerances if the three-quarter dimension had been held closer. Cost of the operation would have been increased however. Reducing tolerances below part print tolerances is not a good practice
2. If the three-quarter inch dimension was shown on the part print instead of the one half inch dimension, the first process would have been correct.

Figure 206. Elimination of Tolerance Stacking

sketch helps the process engineer to maintain control of his operation sequences.

Information on the operation sketch includes:

1. A sketch of the part as it appears after the operation. It involves one view, two views, or a cross-section, depending on requirements.
2. Operation description.
3. Operation number.
4. Volume of metal removed; the surface worked on is sometimes indicated by cross-hatching.
5. The dimension produced at the operation.
6. Position of locators, supports, and clamps.
7. Machine or equipment name and number.
8. Production department and number.

A typical set of operation sketches is shown in Figures 207, 208, and 209. Operation sketches should be made for any operation involving metal removal, change in contour, surface treatment, or inspection. Sketches are not necessary for washing, receiving inspection, painting, and other such operations.

Advantages and uses of the operation sketch are:

1. Reduces errors caused by improper placing of locators.
2. Reduces chances of missing operations needed to make part. The operation sketches can be checked against the part print to insure that all dimensions have been accounted for.
3. As an aid to the die designer for placement of locators, supports, and clamps.
4. As an aid to the production operator and foreman. The sketches tell exactly what is to have been completed prior to an operation in question. To obtain such information from the operation routing requires more time.
5. The chief process engineer may use the operation sketch to check on the soundness of the processing and location system.
6. As an aid to truck drivers, stock clerks, and others to help in identifying parts and in checking the completeness of the parts.
7. When part print and engineering changes come through, the sketches aid in determining what revisions will be necessary in the tooling and equipment.

The sketch may be drawn by the process engineer or by a draftsman working under his supervision. The sketches would be duplicated in the same manner as die drawings.

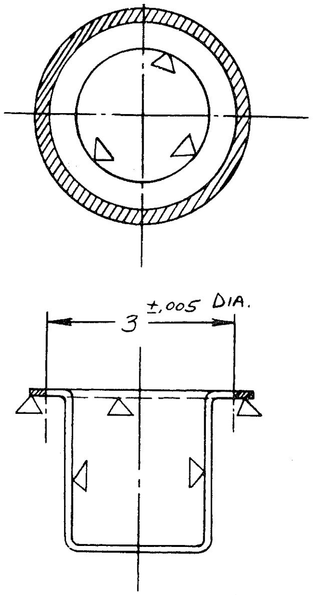
The more operations required to make a part, the more important it is to have operation sketches. When the part is made completely in a progressive or transfer die, operation sketches are not necessary. The sketches are more useful when single-station dies are being used.

OPERATION DESCRIPTION ------- **Trim cup to 3.000 ± .005 diameter**

OPERATION NO. 50

PRESS NAME ------**Bliss #28 OB1**

PRESS NO. 23697



DEPARTMENT ------**Small parts**

DEPARTMENT NO. 35

PROCESS ENGINEER ______________ **Date** ________

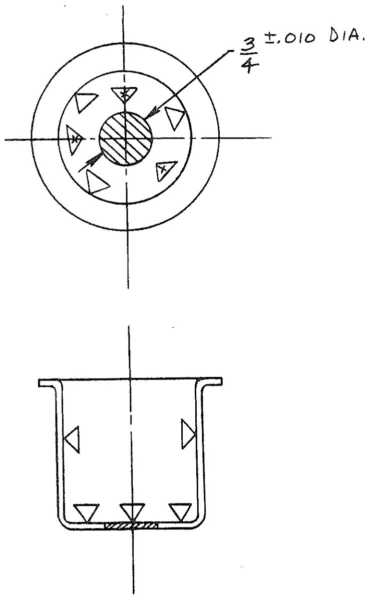
Figure 207. Typical Operation Sketch

OPERATION DESCRIPTION ------- Punch .750 ± .010 diameter hole

OPERATION NO. 60

PRESS NAME ------- Minster S2-100-48-24

PRESS NO. 12583



DEPARTMENT ------- Small parts
DEPARTMENT NO. 35

PROCESS ENGINEER ______________________ Date ________

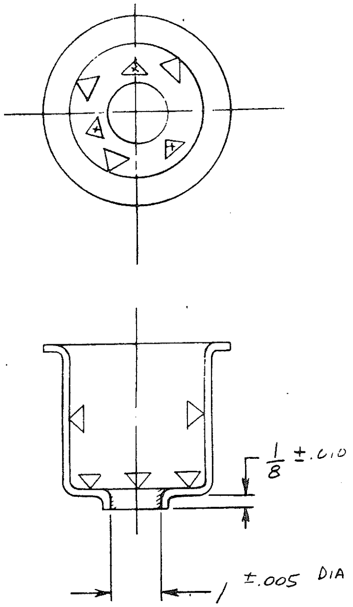
Figure 208. Typical Operation Sketch

OPERATION DESCRIPTION -------- Flange hole to 1.000 ± .005 diameter with height of .125 ± .010

OPERATION NO. 70

PRESS NAME ------ Cleveland S1-50-36-24

PRESS NO. 24590



DEPARTMENT ------- Small parts
DEPARTMENT NO. 35

PROCESS ENGINEER ______________________ Date ________

Figure 209. Typical Operation Sketch

Selection of Tooling and Gages

After the sequence of operations has been determined and the operation sketches with locating points have been completed, the process engineer must decide what tooling is necessary. This step will decide, to a great extent, the cost of producing the part. Actually, the style of tooling to be used was determined at the time that the operation sketches were made. For example, if the sketch shows a blanking and punching operation occurring at the same time, a compound die is to be used. If a progressive or transfer die is to be used, a layout of the progressive strip or steps is made instead of operation sketches.

Many of the rules and principles to be followed in tool selection have been discussed in other chapters. For sheet metal work, selection of tooling consists mainly of deciding on style of die construction. If operations are to be combined into a single die, for example, several construction styles are available. The following characteristics will aid in the selection of dies:

Cutoff Die

1. Low die construction cost.
2. High percent utilization of sheet metal.
3. Easy to set in press.
4. Shut height not usually critical. No bottoming in die.
5. Easy to resharpen.
6. Fewer die components and less maintenance.
7. Limited to blank shapes that nest perfectly.
8. Used to cut up scrap skeleton in a blanking or progressive die.
9. High production rate of blanks per hour is possible.
10. Edge of blank is more distorted because metal moves as it is cut.
11. Blanks or scrap usually disposed via chutes. Drop-thru ejection used on special occasions.
12. Cutting on one edge of punch and die steel causes severe deflection of steel. Height to width ratio is critical.

Drop-Thru Die

1. Suited for blanking operations where perfect nesting is prevented by the blank shape.
2. Blanks or scrap ejected by dropping through die steel and into a chute in die shoe or through press bed into a pan or conveyor.
3. High production rate possible due to the efficient ejection system.
4. Limited to small- and medium-size parts. Hole in press bed not usually large enough to permit passage of larger parts.
5. Die cost is higher than for the cutoff type.
6. Blank or slug is distorted into a dish shape.
7. Small projections in die steel cutting edge make drop-thru of blank more difficult.
8. Die steel must be on the die shoe.

Return-Type Die

1. Necessary when part cannot be ejected by drop-thru principle because there is not a sufficient hole in press bed.
2. Part is held fairly flat. Knockout plate may be bottomed to flatten blank.
3. Die cost is higher than drop-thru style.
4. Ejection of part is difficult. Requires use of inclined press, shovel unloader, air blast, kickers, and other special ejection systems.
5. Production rate is slower due to method of ejection.
6. Die steel may be placed on punch shoe to utilize the solid knockout device.
7. Used extensively for large panel dies for flanging and forming. Most forming and drawing dies use the return principle.

Compound Die

1. Close tolerances can be maintained between holes and blank edge, on concentricity dimensions, and between cut and formed or drawn surfaces. Greatest accuracy possible with this die because part is not relocated between operations.
2. Die makes efficient use of press and operator. One setup replaces two or more dies, presses, and operators.
3. Die cost is high due to number of components and compactness.
4. Die resharpening and maintenance difficult.
5. Low production rate and difficulty in part ejection, as was true with the return-type die. Drop-thru principle cannot be used.
6. Because many components must be squeezed into a limited space, use of a compound die is not always practical. This is determined greatly by the part size and spacing of surfaces.

Combination Die

1. Used extensively when right and left hand parts are required. When size permits, a combination die is used at each operation. The die then consists of two stations that are almost identical except that they are reversed.

2. Used to utilize fully the press capacity. Two or more small dies are assembled to a large common shoe. The stations may be for different operations on the same part or for different parts.
3. Production rate is low. The parts are usually loaded manually. Often more than one operator is required per press. All operators must push the palm buttons before the press will cycle.
4. Used mainly for cutting and forming. Combination of drawing with other operations does not work satisfactorily. This is due to the slow drawing speed and blankholder requirements.
5. Ejection of parts is somewhat difficult especially when there there are two stations side by side as well as back to back.
6. Die cost is usually not much higher than for single-station dies to do the same work.

Continental Die

1. Very low production. Sheet metal is placed in die by hand, and the die placed in and out of the press and then forced apart with an arbor press.
2. Very low die cost. No die set is used.
3. No cost for setting die in press.
4. Part distorted in a manner similar to the drop-thru die.
5. Difficult to apply to dies requiring low clearance.
6. Used for very low production and experimental work.

Sub-Press Die

1. Low production rate. Used often in a press brake for punching holes.
2. Higher cost than continental die, since a die set is employed.
3. Stroke of die limited by compression allowed on the springs that open the die.
4. Less time required to set die in press. Die is not attached to the press ram.

Progressive Die

1. Very high die cost. Consists of many die components that are compact. Extra expense in locating the strip.
2. Very high production rate. Sheet metal is fed automatically and part ejected into container. Press is run continuously with no lost time for loading and unloading parts.
3. High equipment cost. Requires special press attachments, feeds, cradles, and reels.
4. Limited to small- and medium-size parts.

5. Not readily applicable to irregular-shaped drawn parts requiring variations in blankholding pressure and contour. May be used for cupping and box drawing.
6. May be used for both thin and heavy sheet metal.
7. Each station or operation is limited by the adjoining operations. The parts are attached together by the skeleton and the skeleton cannot be severely stretched.
8. Some difficulty encountered with soft and thin sheet metal. Pilots tend to distort hole rather than shift strip into location.
9. Die setting, resharpening, and maintenance are difficult.
10. Coiled stock is used. To hand feed strip stock would require too precise timing on the part of the operator. Special feeding devices are available for strip stock.
11. Part cannot be turned over or around, thus limiting some operations.
12. Higher scrap produced because parts cannot be nested to best advantage.

Transfer Die

1. Very high production rate. Sheet metal in coil form or blanks is fed automatically and parts are ejected into container.
2. Very high die cost. Many die components, but they are usually not as crowded as in the progressive die.
3. High equipment cost. Requires special attachments and feeding device on press.
4. Not applicable for irregular-drawn panels for the same reasons as the progressive die. Because the parts are fed mechanically through the die, slightly larger parts can be made than with the progressive die, which relies on a narrow sheet metal ribbon between parts.
5. Greater versatility than with the progressive die. Parts are not tied together. Each station is free of the next.
6. Greater accuracy than with the progressive die. The mechanical feed is more positive than a roll feed plus pilots.
7. Not easily used for thin sheet metal. Fragile or flimsy parts would be distorted when gripped by the fingers for feeding.
8. Die setting, sharpening, and maintenance costs are high.
9. Blanks may be cut out in one press to obtain the highest percent utilization. The blanks are then fed with a magazine feed into the transfer die and press.

Shuttle Die

1. Used especially for large and flimsy parts. Shuttle lifts and transfers and does not grip the part, thus avoiding distortion of the part.

2. May be used to transfer part from station to station in the die as well as from press to press.
3. Production rate usually slightly lower than progressive or transfer dies.
4. Die must be cut away to provide space for shuttle bars. Often die details must be mounted to the bars. Not applicable to a draw die for these reasons.
5. Usually easier to set up and maintain than the progressive or transfer dies.
6. Equipment cost is high.
7. The most positive method of loading and ejecting large parts. Can be used to completely automate a press line.

As seen by the long list of considerations, the process engineer must make a detailed study if the most economical die is to be selected. Also, getting the production in the time required is a major criteria. The process engineer's choice at this time will greatly affect the tooling cost and over-all production costs. This is certainly a critical step in processing.

The major consideration in selecting gages is that if standard gages can be purchased, cost will be considerably reduced. The designing of special gages and inspection in general is a specialized field of its own. It is recommended that special books on gaging be consulted by the process engineer.

Selection of Equipment

The primary concern here is the selection of presses. Sometimes a press is purchased new for a particular die. More often, a press is selected from those available in the plant. Again, most of the knowledge required has been discussed in other chapters. The major considerations in press selection are:

1. Tonnage
2. Speed or Strokes per Minute
3. Bed Area
4. Shut Height and Adjustment
5. Stroke
6. Number of Presses Required

Since the die is designed after the press has been selected, the process engineer must estimate the die requirements. He must calculate or estimate what tonnage is required to operate the die, the approximate shut height, and the area in the plan view. To have the ability to do this, the process engineer should have some experience in die design and construction. He need not be an expert in these fields, however.

Tonnage. The tonnage required by the die may be estimated with the formulas presented in previous chapters. The position at which the tonnage is required is a critical factor. Presses are rated in tons of capacity near the bottom of the stroke. The capacity decreases considerably near midstroke. A typical stroke-tonnage chart is included in the appendix. Notice that this chart is for double-geared presses. The length of stroke as well as position of stroke both affect the tonnage reduction above the bottom position. Similar charts should be obtained from the press manufacturers for all presses used in the plant.

On the stroke-tonnage chart, the example (500) ton press with a (32) inch stroke has a capacity of only (220) tons at a point three inches up from the bottom. This example may be used to illustrate use of the chart as follows:

Press Capacity	*Inches up from Bottom of Stroke*
220 tons	3
260	2
365	1
500	½

If a cutting operation is being performed one-eighth of an inch from the bottom of the stroke, the press can be loaded to the full capacity of 500 tons. Therefore, the chart is not required, since the rated tonnage may be used.

If a forming operation is being performed one inch above stroke bottom, the press can be loaded to (365) tons. The position at which forming begins is important. If the form die bottoms, then the bottoming force must be compared to the full press capacity. Often, it is wise to make checks at both points in such a die.

If a drawing operation is being performed on a cup that is three inches deep, the press can be loaded to (220) tons. During drawing of this cup, the maximum force occurs three inches above stroke bottom. Selection of press capacity for drawing dies is the most critical, since drawing starts a great distance up from stroke bottom.

Ram speed. The speed or velocity of the ram at the time that the die starts to work the sheet metal is often important. Cutting and forming dies can usually be run at high speeds ranging from (100) to (500) strokes per minute. With such speeds, the production rate is high. A ram velocity versus distance from stroke bottom is included in the appendix. Notice that the ram approaches zero velocity as it nears bottom of the stroke. Therefore, the velocity does not change appreciably as the strokes per minute increase when cutting near the stroke bottom. Also the speed of cutting is not as critical as that of drawing. The velocity at which the

80540

punch steel contacts the blank and the velocity of drawing are critical. Failure results when high speeds are used. The deeper the drawn part, the higher will be the contact velocity for a given press. Drawing presses and dies run between 5 and 50 strokes per minute. The presses having longer strokes have fewer strokes per minute. Press speed should be as high as possible to obtain high production rates. Press speed is limited by the operation and distance of die contact above stroke bottom.

Bed area. The press bed and ram areas must be large enough to accommodate the die. The processor must estimate the room required for springs, stripper plate, blankholder, pad, keepers, and gages as they are placed about the contour of the punch and die steels. The punch and die steels will be the size of the part. Larger parts require more area for these accessory details. Some die and punch shoe space should be left outside of the keepers to permit room for bolts to attach the die to the press. The processor must rely on his knowledge of die design to predict the bed area required.

Shut height and adjustment. The processor must estimate the shoe thicknesses, die steel height, and punch steel height to obtain the approximate die shut height. The press shut height with adjustment must be suitable for the die. If only a press with too much shut height is available, then a bolster plate or parallels must be used. Again, the processor relies on his knowledge of die design techniques that are discussed in following chapters.

Press stroke. The press stroke is important when considering ejection of the part. Several factors are:

1. Blanking dies and dies cutting on flat parts should be run in pressses having a short stroke. Two to six inches of stroke is sufficient to allow removal of the flat parts.
2. Some forming dies require longer strokes, since the die must open to allow removal of the part. If a flat part has been formed to a shape that is difficult to remove when a short stroke is used, sometimes the formed part can be removed by sliding it out sideways. Then a shorter stroke may be used.
3. For draw dies, the die must open so that the part can be removed. For example, if a three-inch deep cup is drawn, the space between open die halves must be at least four inches or more. Draw dies require longer strokes.

As short a stroke as possible should be used, for presses with short strokes can be run at higher speeds, thus increasing production. The main limiting factor on stroke is the height of the part and the ejection system used.

Number required. In general, when a high production is required, the processor selects a die of the multi-station type which will achieve this production rate. When many large parts are to be made, the multi-station die is often not applicable, and single-station dies must be used. In this case, it is then necessary to have several duplicate dies and presses for the same operation. The processor must estimate the number of duplicate dies and presses required. In extreme cases, entire press production lines are duplicated to obtain the production rate desired.

Operation Routing

To convey information to others, the process engineer originates the operation lineup or routing. Tool routing is another term used for this operation. The following information is included on the routing:

1. Operations
   a. Operation number
   b. Sequence of operations
   c. Operation description
2. Tooling
   a. Tool numbers
   b. Dies and descriptions
   c. Gages and descriptions
3. Accessories
   a. Gloves
   b. Aprons
   c. Tongs
   d. Face Shield
   e. Towels
4. Supplies
   a. Lubricants
   b. Paint
   c. Rust preventative
5. Equipment
   a. Press name and number
   b. Press model and description
   c. Cradle or reel and numbers
   d. Feed
   e. Straightener
   f. Roller leveler
   g. Washer, degreaser, tumbler, etc.
   h. Mill, drill press, riveter, etc.
6. Part
   a. Name and description
   b. Number
7. Department name and number
   a. Production
   b. Inspection

PROCESS — MACHINE AND TOOL ROUTING

PART NAME		MATERIAL
PART NO.	MODEL	PURCHASED AS
TOTAL QUAN	PCS. PER LOT	TREATMENT

LOCATING POINTS △ ▽ △	SUPPORT POINTS F A	HOLDING FORCE →	FINISH

OPERATION DESCRIPTION

CHECKED FOR SAFETY BY DATE

GENERAL MOTORS INSTITUTE — FLINT, MICHIGAN	NO. OF SHEETS	SHEET NO.

MACH. NAME		WRITTEN BY	
G.M. CLASS. NUMBER	TAG NO.	DATE	
SPEED RANGE	FEED RANGE	CHECKED BY	DATE

SPECIAL TOOLS	TOOL NO.	QUAN.

COMMERCIAL STD. TOOLS	G.M. TOOL NO.	QUAN.

STD. AND SPECIAL GAGES	GAGE NO.	QUAN.

PROCESSING - - MACHINE AND TOOL ROUTING	General Motors Institute - Flint, Michigan	No. of Sheets	Sheet#
Part Name	Material	Total Quan.	Tool Eng.
Part No.	Purchased As	Pcs. Per Lot	Date

Oper. No.	Process Sheet No.	Operation Description	Machine Name	Mach. No.	Special Tools	No. Req.	Gages Special & Commercial	Gage No.	Commercial Tools	No. Req.

8. Routing date
9. Number of sheets and sheet number

The operations are usually numbered 10, 20, 30, 40, and so forth so that new operations may be inserted at later dates without changing previous operation numbers. Typical operation routing forms are shown on the following pages.

As tools or operations are cancelled, or new tools added, the processor revises and distributes the pages necessary to bring the routing up to date. A column is often provided to note if the tool is new, cancelled, or has been changed. Often, the routing and operation sketches are combined as shown.

Commercial tools and gages are those which may be purchased as standard catalog items. These tools are listed separately from those made specially for the operation.

Orders

To carry out his responsibilities and function, the processor originates several orders, as follows:

1. Order to design special tools and gages.
2. Order to build special tools and gages.
3. Order to purchase commercial tools and gages.
4. Order to purchase equipment.
5. Order to cancel special and commercial tools and gages.
6. Order to change design of special tools and gages.
7. Order to change special tools and gages.
8. Order to reinstate special tools and gages.
9. Order to build additional special tools and gages.

These are really requests for a purchase order, which will be originated by the purchasing department.

A primary function of the processor is to approve all designs before production commences. Generally, he also obtains the approval of the production or inspection foreman who will use the tools or gages.

Engineering Changes

The engineering change is originated by product engineering to change the part print or production specifications. The result is an engineering change release and changed part print. Reasons for making an engineering change may be one or more of the following:

1. Reduction or increase in sales.
2. Product failure in service.
3. Customer demands on styling, color, etc.
4. Manufacturing cost too high.

5. Unavailability of material.
6. Manufacturing difficulties.
7. Assembly difficulties.
8. Errors on original part print.
9. New lower cost or better material now available.
10. Model change.

Often, when difficulties are predicted or occur during manufacturing, the process engineer originates a request for an engineering change to correct the situation. With the request, he submits a part print that is marked to show the exact change desired. This difficulty in manufacturing will increase production cost and decrease production rate. Often the manufacturing difficulty is called to the processor's attention by the production supervisor.

If the processor discovers an error on the part print, he may request an engineering change.

Followup

Another function of the process engineer is to "followup," to see that all tools, gages, and equipment are functioning properly. When necessary, he originates orders to add tools, cancel tools, or change tools to correct any difficulties. Often minor changes are made on tools by the production department or tool room. The foreman should notify the processor of such changes so that he can have the tool drawings brought up to date. If drawings are not corrected, various difficulties could occur, as follows:

1. Duplicate tools might be built, and they would not incorporate the change.
2. If similar tools were designed and built using the older design for reference, the change would not be incorporated.
3. The designer would not realize the error made and would commit the same error again.

This followup function can usually be carried out during the slack period following a model changeover. At this time, all routings are written, tools built, and equipment received. Until the part prints for the next model are released, the processor is busy taking care of engineering changes and changes resulting from followup. Experience gained during this period is used to better the next processing task.

Estimating

Many times a plant must bid or quote on a product, and, if the price is suitable, the plant receives an order for so many units per month or year. To make an accurate estimate of the cost per unit, the following

factors are added:

Material costs per part
+
Direct labor costs per part
+
Tooling costs per part
+
Overhead per part
+
Profit per part

Selling price of product

The plant receives a request for quotation along with the part print. It is normally the processor's task to estimate the tooling costs. Tooling cost includes special tools, commercial tools, and equipment. To make the estimate, the processor relies on experience and records of tooling costs for similar parts made in the past. He must make adjustments to account for changes in costs due to wage variations, economic conditions, and improved tool room techniques.

At best, this preliminary estimate is approximate. If the estimate is higher than actual costs, the plant may fail to get the order. If the estimate is too low, the plant may get the order and actually lose money. Thus, estimating is truly a critical function.

In some plant organizations, the estimating is done by an estimator who works in the tool design and/or construction area. This estimator has the regular task of estimating the cost to build tools after the design is complete. In some instances, he may even estimate the cost to design the tool.